Design and Performance Curve Generation by CFD Analysis of Centrifugal Pump

Supriya Jadhav[1], Vaibhav Ghodake[2], Pavan Chipade[3], Shubham Gaikwad[4], Vikram Ghule[5], Nilesh Gaidhani[6]

[1]Supriya Jadhav, UG student, Dr. D Y Patil School of Engineering, Pune
[2]Vaibhav Ghodake, UG student, Dr. D Y Patil School of Engineering, Pune
[3]Pavan Chipade, UG student, Dr. D Y Patil School of Engineering, Pune
[4]Shubham Gaikwad, UG student, Dr. D Y Patil School of Engineering, Pune
[5] vikram Ghule, Professor, Dept. of mechanical Engineering, Dr. D Y Patil School of Engineering, Pune
[6]Nilesh Gaidhani, Design Executive Indo Pump, Pune.

---***---

Abstract - *This work investigates a systematic numerical approach that employs Computational Fluid Dynamics (CFD) to obtain performance curves of a backward-curved centrifugal pump. Capacity curve obtained from the CFDS analysis of centrifugal pump gives wide approach to parameters such as cavitation and reverse flow. Semi open impeller with single volute casing defines the path of flow. This study is focused on effect varying discharge on its performance parameters such as head, required power and efficiency. Head discharge relation will give ease of pump selection. Mesh generation technique had discussed in the project work for better CFD results.*

Key Words: *pump design, construction of blade, CFD analysis, cavitation, performance curve.*

1.INTRODUCTION

A pump is a mechanical device for moving a fluid from a lower to a higher location or from lower to higher pressure area. Performance of the ump may be affected due to some geometrical and input parameters such as blade angle, impeller size, discharge and head required. To overcome the problem, designers often change the geometry of the pump selection parameters. Specific speed determines the geometry of the impeller and forces in pump due to fluid flow whole design may be depend on the performance curve generation which helps the overcome the selection problem s and the required BEP. While overcoming the losses the designed modification can be done.

1.1 OBJECTIVES

1. Familiar approach to improve the design of centrifugal pump and optimize its operational parameters.
2. To study the centrifugal pump approaching towards the radial flow pumps.
3. To evaluate pressure distribution at blade and shroud region of the centrifugal pump.
4. To obtain the optimum pump impeller design for effective suction of pump.
5. Plot performance curve of pump (H&Q curve, Efficiency vs. discharge vs. head curve).
6. To investigate the effective of impeller geometry on pump performance.
7. To analyze the effect of variable discharge on cavitation pressure counters at impeller blade.
8. To check the performance of pump such as efficiency, hydraulic power output wrt varying discharge.

1.2 METHODOLOGY

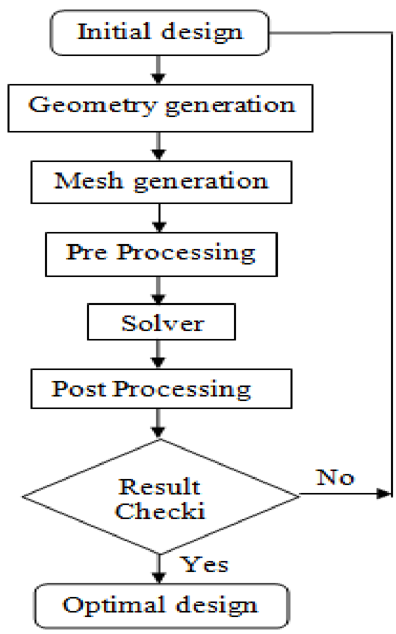


Fig.1 flow of research

2. DESIGN AND DEVELOPMENT

2.1 Impeller

Pump have to carry 1000m^3/hr with abrasive material for the total head of 75 m from the storage tank at atmospheric pressure. Specific gravity of fluid flowing through the pump is 1.0. Pump having the speed for impeller is 1450 rpm. The overall efficiency is assumed to be 80% for the pump. While designing the pump, design will tends towards the missed flow region. While designing the pump modification in the design may vary with respect to results.

Table -1 Dimensions of Impeller

Parameters	Dimensions
Outside diameter (d_2)	530 mm
Eye diameter (d_{eye})	288 mm- 290
Vane inlet edge diameter (d_1)	289mm
Outlet width (b_2)	44 mm
Inlet width (b_1)	81 mm
Diameter of shaft (d_{sh})	59 mm
Vane inlet angle (β_1)	17^o
Vane outlet angle (β_2)	22.5^o
Vane thickness (t)	8 mm

Genration of impeller blade is mainly focused on the blade curveture. Bl;ade curveture the pump performance as it seprstes pressure regions. Blade curve is genrated by multiple arc method.in multiple arc method, impeller is divided into 6-8 curves. Final blade curve genrated is replica of the curves in each section. Also impeller shroud is designed with inlet and outlet width dimensions for smooth flow of fluid.

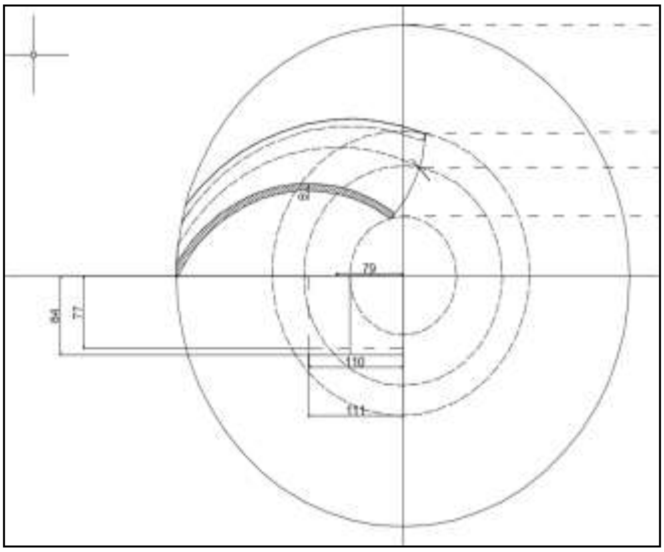


Fig.2 Blade curve by multiple arc method

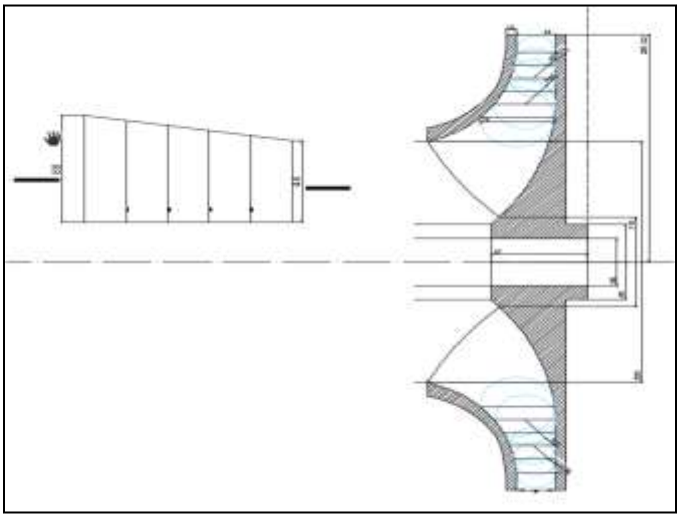


Fig.3 Developement of shroud

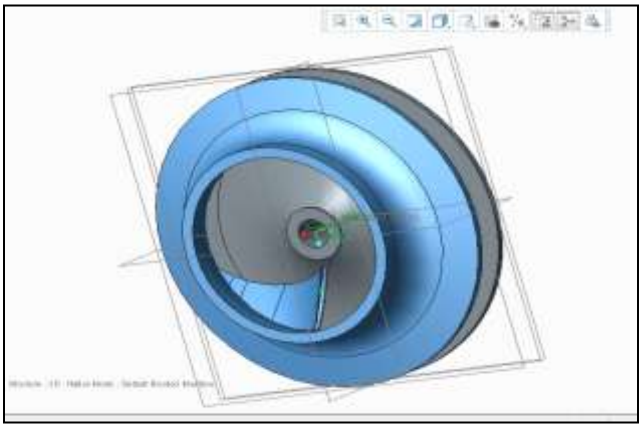


Fig.4 Developement of Impeller Blade and Shroud

2.2 Single Volute Casing

Volute of centrifugal pump is generated in software CF Turbo. CF turbo is popular tool for impeller and volute generation due to Rapid design of hydraulic high-quality pumps. Integration of established pump design theory. Comfortable, reliable and user friendly Direct interfaces for many CAE-software packages .Comprehensive and detailed documentation manual.

Table-2 Input parameters for volute in CF Turbo

Required discharge	1000 m^3/hr
Head required	75 m
Inner diameter	557 mm
Inner width of volute	90 mm
Outlet nozzle diameter	250nmm
Neck of volute	23^0

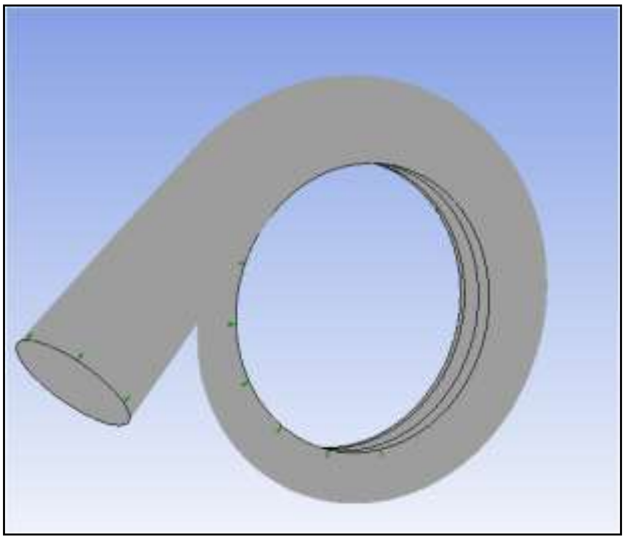


Fig.4 Generation Single volute in CF turbo

3. MODIFICATIONS OF STATOR COMPONENT:-

3.1 In-pipe And Elbow:

In pipe is used to guide the inlet water to the elbow. Elbow is intermediate part between the Inpipe and impeller shroud. Fig (2). Interference is created between the Inpipe elbow and impeller in CFX pre.

3.2 Out-pipe

Function is to guide the flow of water coming out from the volute casting. This is required to minimize the whirl component and to achieve better results.

Table-3 dimensions of auxiliary components

Element	Diameter	length
In Pipe	250	1000
Elbow	250-290	175
Out pipe	250	500

Interference is creates between the Inpipe and Elbow is needed during CFD simulations, so the dimensions must match with other component. Outpipe coming out from the casing outlet is increasing in cross-sectional area to create more head at outlet by converting the kinetic energy into pressure.

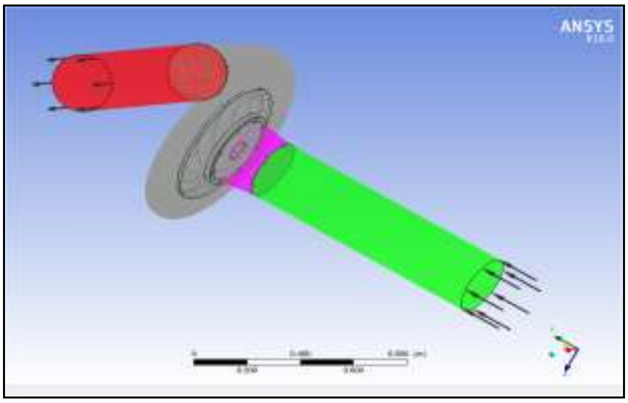


Fig.5 auxiliary component Inlet pipe, Elbow, Outlet pipe

4. CFD ANALYSIS

As the impeller moves through the fluid, low-pressure areas are formed because the fluid accelerates around the blades. The higher the fluid velocity, the lower becomes the local pressure. If it falls below vapor pressure, the fluid vaporizes and forms small bubbles of gas. These are dragged to areas of higher pressure, where they collapse and can cause very strong local shockwaves in the fluid, which may even damage the blades. CFD helps to design pumps with favorable cavitation behavior over a wide operating range.

i) Preparation of surface model of the impeller and Casing using software like PRO-E, CATIA, Uni-Graphics.
ii) Grid generation by using software like ANSYS-IECM.

iii) Application of boundary conditions using software like CFX PRE.
iv) Solution and analysis of results using software like ANSYS-CFX POST, Fluent.
v) Analysis of flow through hydraulic passages and prediction of pump performance
characteristics by application of computational fluid dynamic (CFD) techniques.

4.1. MESHING

The subdivision of the domain into no of smaller, non-overlapping sub-domains: a grid (or mesh) of cells (or control volumes or elements).For the meshing of impeller we used hexahedral mesh and for the volute we used tetrahedral mesh.

Meshing plays main role in the outcome of the cfd results. fine mesh is required near the region where more chances of turbulence, cavitations such as impeller blade wall, volute tongue and low pressure regions. Different meshing techniques can be used to generate the mesh to different parts of pump. Also the condition of surfaces and quantity play main role in mesh quality. For the same degree of polynomial the finite element space generated by hexahedral elements is richer than the space generated by tetrahedral elements. However the tetrahedral elements are best to model complex geometry domain with little distortion of mesh. Moreover, the computational cost for assembling the global stiffness matrix for tetrahedral elements is lower because there is not necessary numerical integration.

Table-3 The mesh statistics is given bellow for each component:-

Sr. no	Component	Mesh type	Node point	Element No.
1	Impeller(Rotating Domain)	Hexahedral	2308800	2438112
2	volute	Tetrahedral	564769	1920449
3	In pipe	Hexahedral	237600	244900
4	Out pipe	Tetrahedral	34435	93026
5	Elbow	Hexahedral	408988	398634

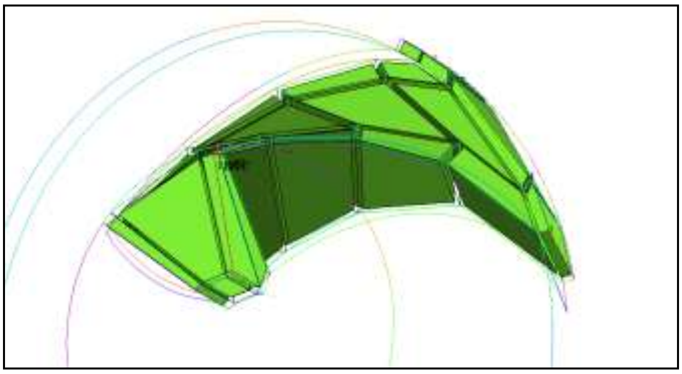


Fig.5 Blocking of impeller

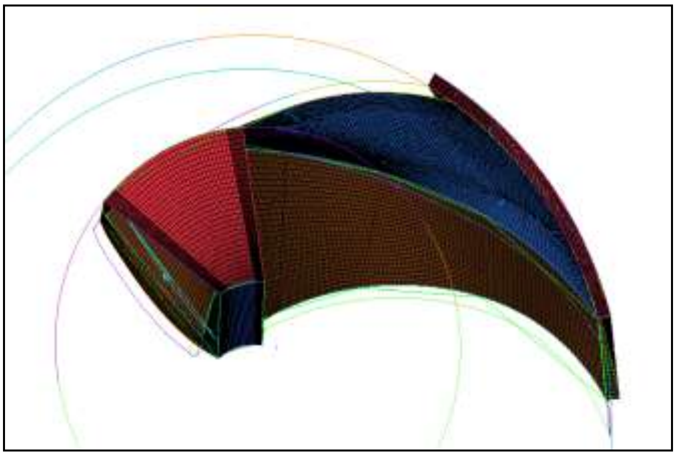


Fig.6 Hexahedral mesh for single fluid passage

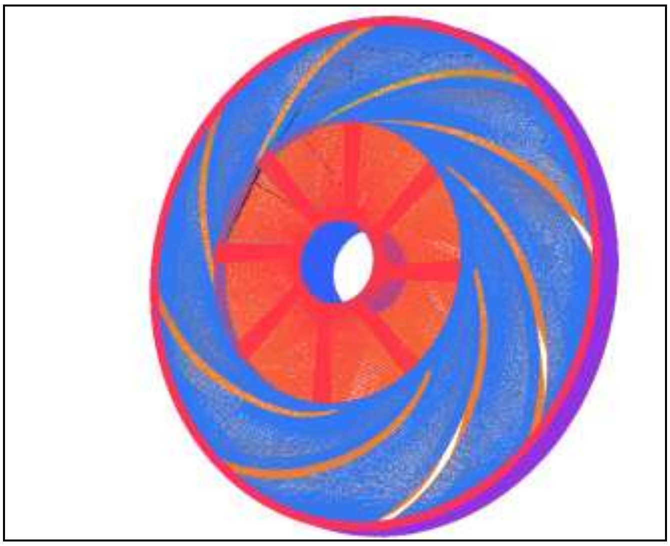


Fig.7 Hexahedral mesh by blocking method

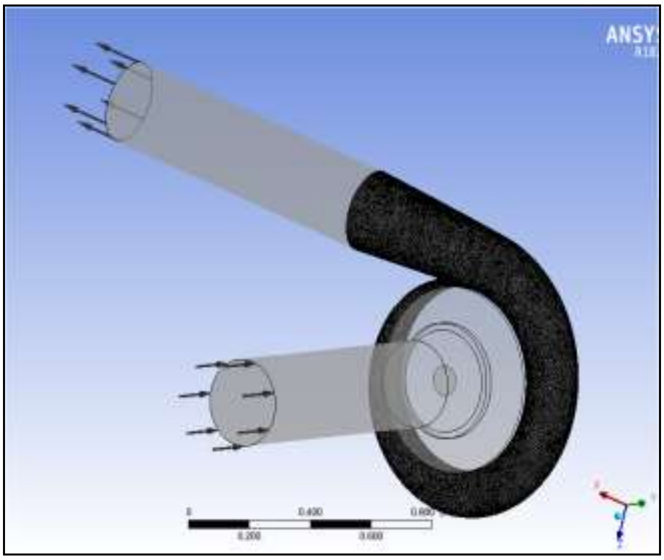


Fig.7 Tetrahedral Meshing of volute

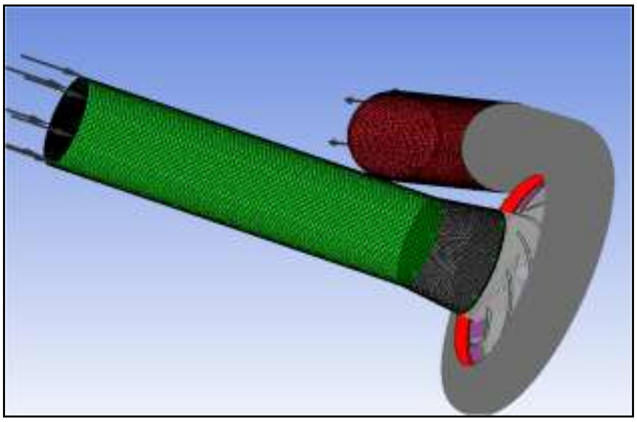


Fig. 8 Meshing of In pipe, Out pipe and Elbow pipe

4.2 Boundary Conditions

Pre-processing consist of the input flow problem to cfd program by means of operator friendly interface and subsequent transformation of this input into a form suitable for use by the solver. The user activities at the preprocessing stage involve:

1. Selection of physical and chemical phenomenon that need to be modelled.
2. Definition of fluid properties, fluid is water hence select the water as fluid.
3. Specification of appropriate boundary condition at cells which coincide with or touch the domain boundary. We give boundary condition as the flow rate of 1000 m^3 /hr, and speed of 1450 rpm as per our problem statement. Inlet discharge conditions vary in 250, 500, 750, 1000,1250, 1500,1750 m^3/hr.
4. Boundary conditions given to the pump are depends upon the type input data and desired results.
5. Input to impeller is given in mass flow rate condition in kg / sec.
6. For rotating domain in pump, wall are consider as the no slip wall and coordinate frame as the rotating.
7. For stator domain, wall is considered as No slip wall and wall roughness consider as smooth wall.

Table-4 Boundary condition given to CFX Pre

Fluid	Water (20^0)
Inlet condition	101353 Pa
Mass flow inlet	277 Kg/sec
Domain motion (Angular)	-1450 RPM
Interference model	General Connection
Interference type	Fluid- Fluid
Mass and momentum option for interference	Conservative Interference flux
Analysis type	Steady state
Turbulence model	Shear stress transport

5. RESULTS

Visualization of the counters of pressure and velocity variation are presented bellow. This contours give the better understanding of low and high pressure areas which are useful in modification in geometrical parameters.

By solving different expressions in the cfx solvers, we can get pump performance parameters such as power, discharge , head obtained, and efficiency of pump on both impeller and pump side.

Table-5 Result table at 1000 m³/hr

Mass flow rate	277 kg/sec
Total pressure inlet	-799659 [Pa]
Total pressure outlet	17193.4 [Pa]
Volumetric efficiency	96.4123 %
Impeller efficiency	80.57 %
RPM	1450
Overall efficiency	75.37%
Total head output	83.51 m
Hydraulic power	237751 [W]
Total torque	1982.92 J

5.1 Pressure and velocity distribution for different discharge

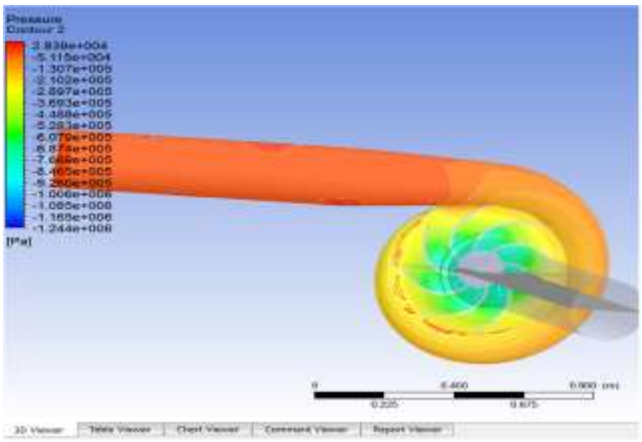


Fig.9 Pressure distribution in the pump volume at 1000 m³/hr

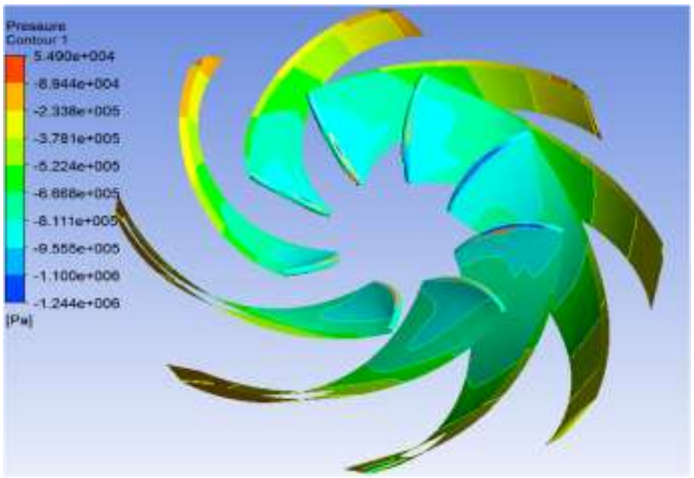
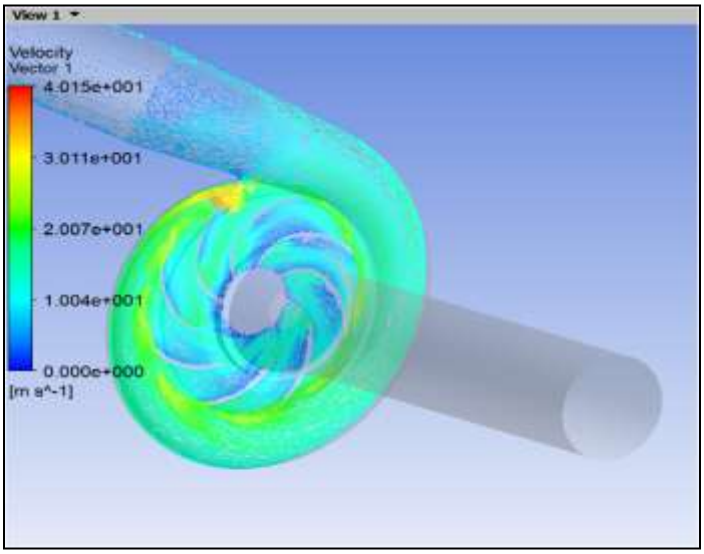


Fig.9 Pressure distribution over the blades at 1000 m3/hr



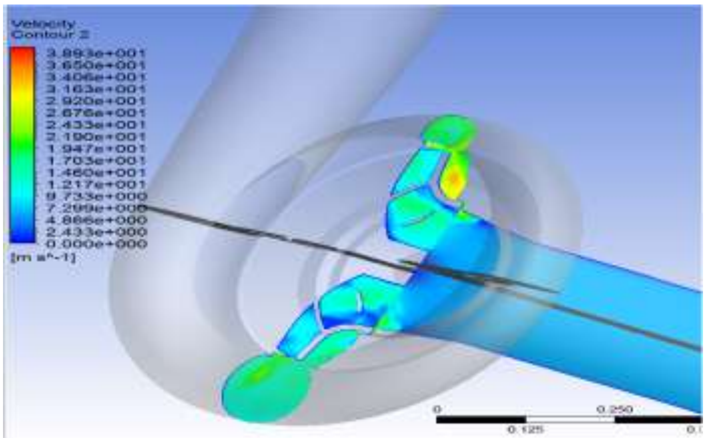


Fig.10 velocity distribution at sectional plane at 1000 m³/hr

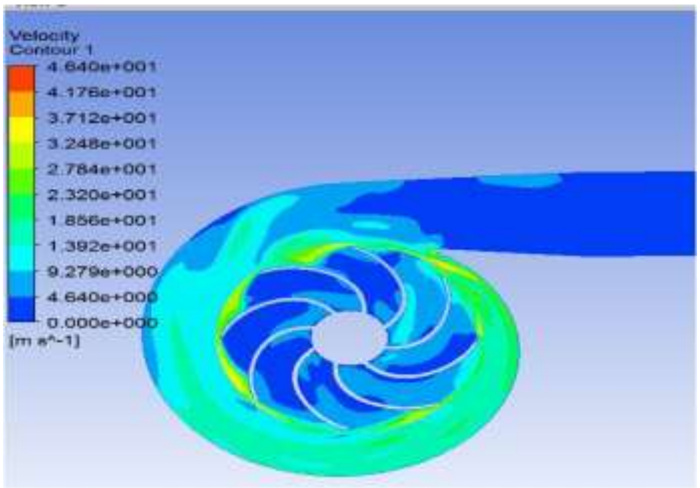


Fig.11 Velocity distribution at 250 m^3 /hr

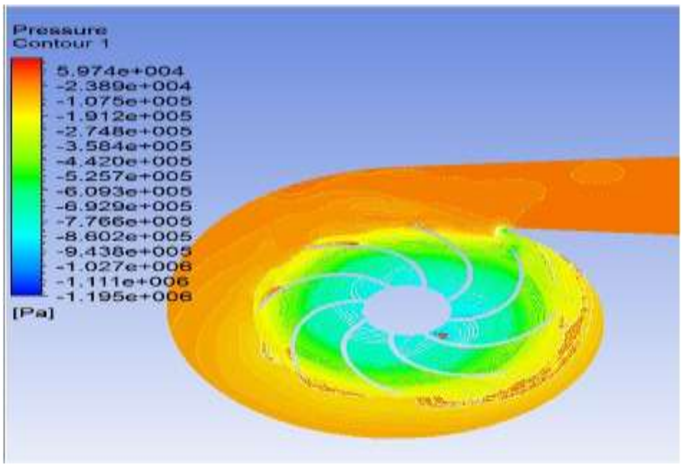


Fig.12 Pressure distribution at 250 m^3 /hr

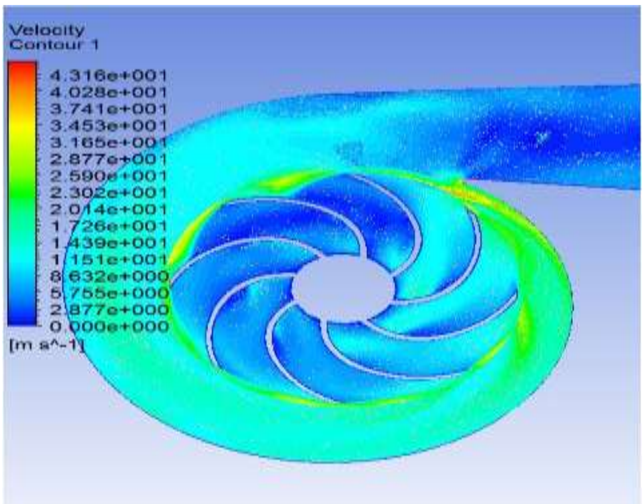


Fig.13 Velocity distribution at 500 m^3 /hr

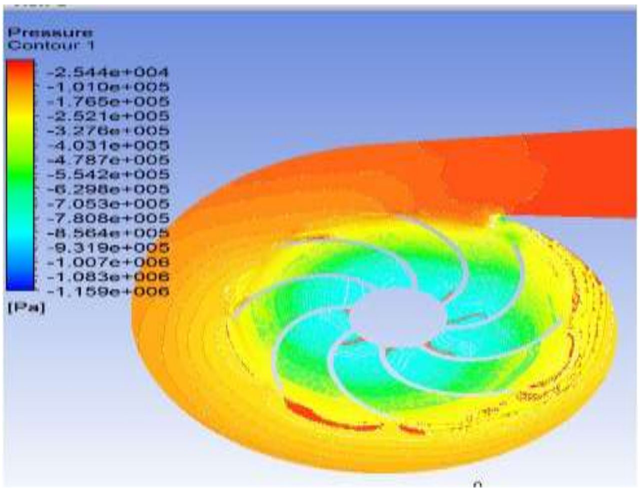


Fig.14 Pressure distribution at 500 m^3/hr

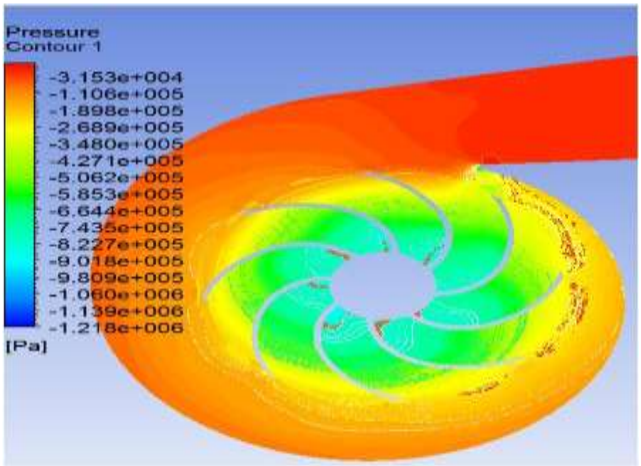


Fig.15 Pressure distribution at 750 m^3/hr

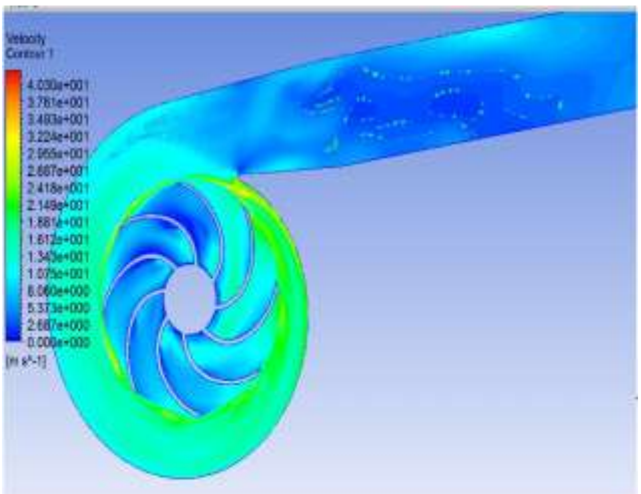


Fig.16 Velocity distribution at 750 m^3/hr

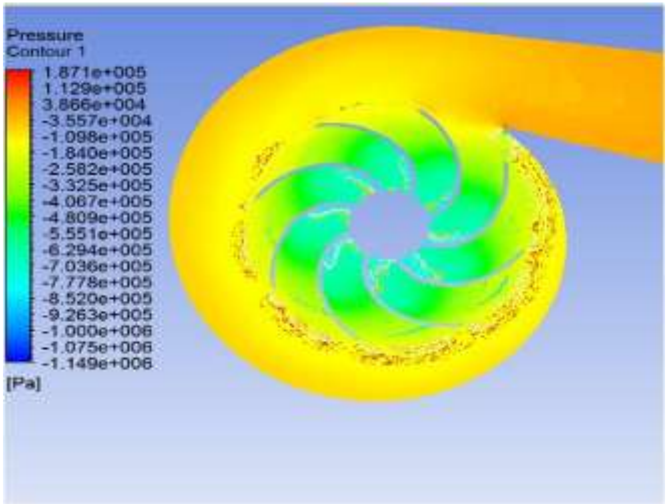


Fig.17 Pressure distribution at 1250 m³/hr

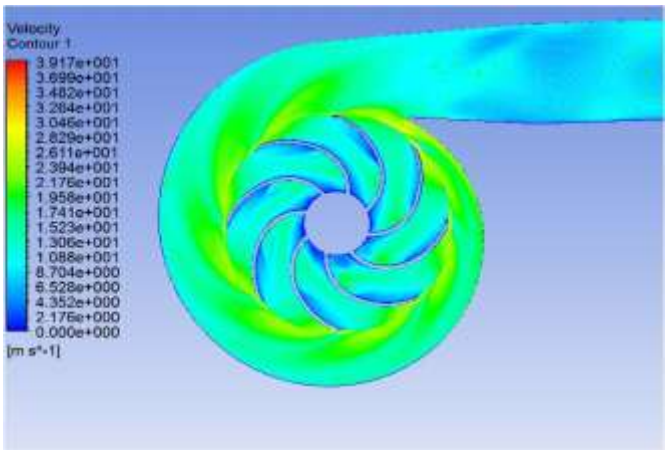


Fig.18 Velocity distribution at 1250 m³ /hr

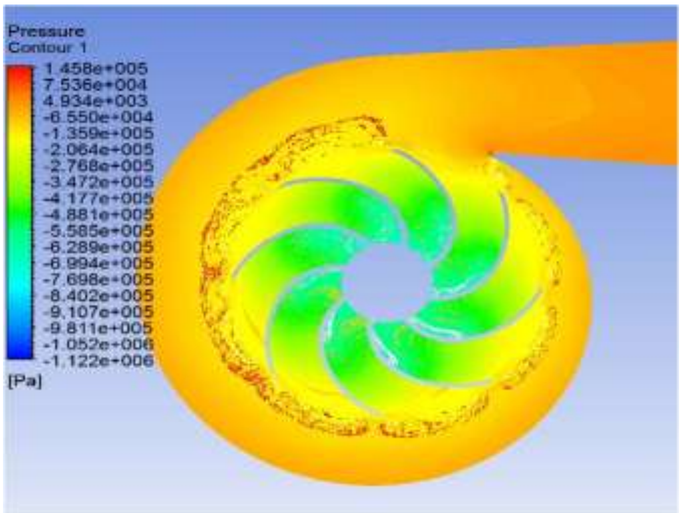


Fig.19 Pressure distribution at 1500 m³/hr

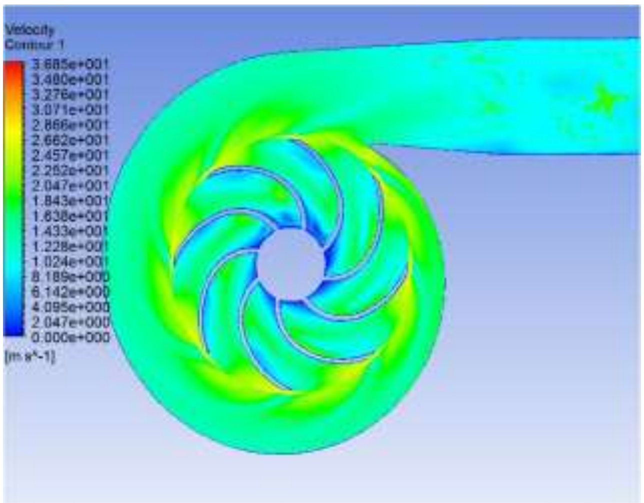


Fig.20 Velocity distribution at 1500 m³ /hr

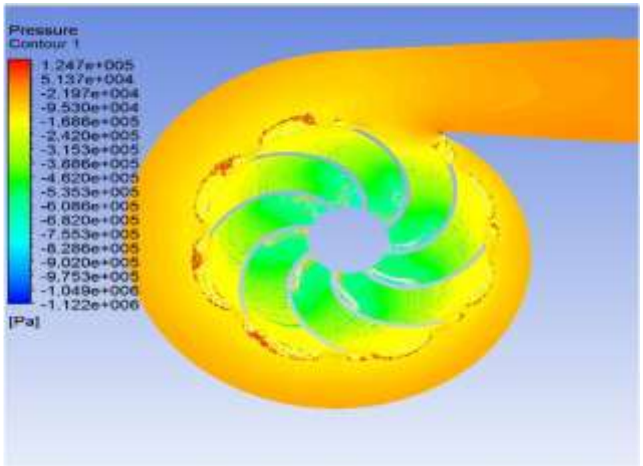


Fig.21 Pressure distribution at 1500 m³/hr

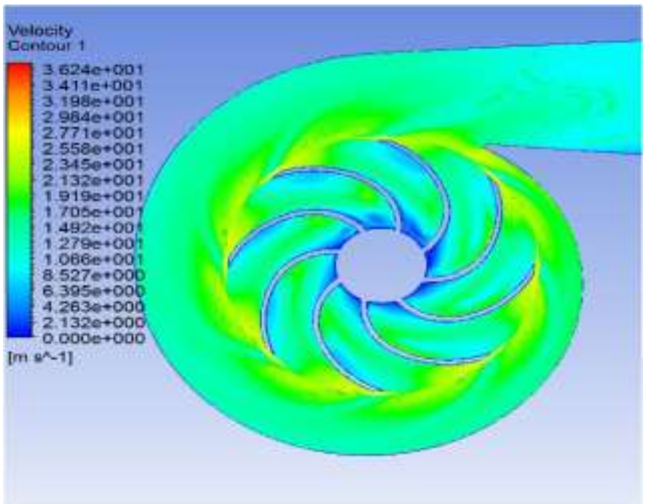
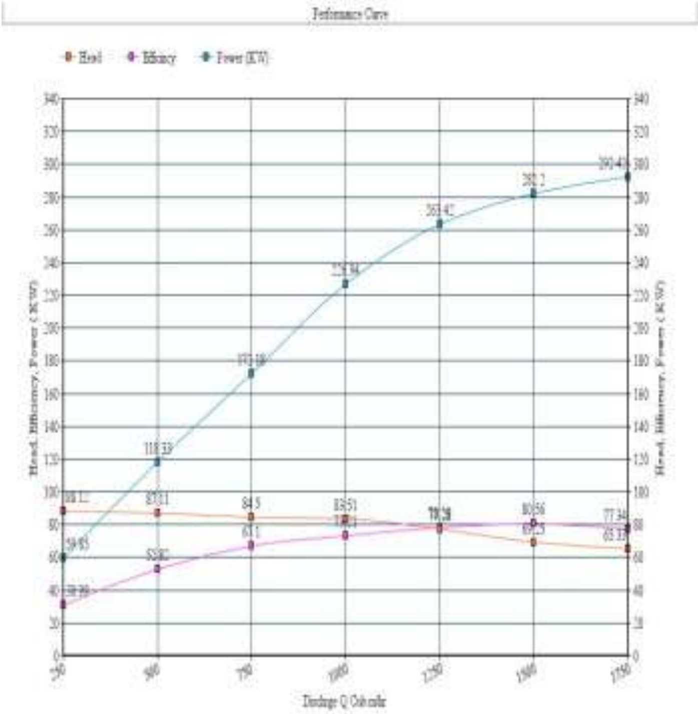


Fig.22 Velocity distribution at 1750 m³/hr



Graph-1 Performance curve

3. CONCLUSIONS

The pump performance calculated by the cfd simulations is well suitable for the given pump requirement. The required head is 77m and designed head of impeller id between 89 m. After simulation of pump, results satisfactory with total head of 83 m , which Is above the required head.

Observations obtained are bellow:-

1. As discharge increases, the power is increases in constant proportion.
2. At low discharge condition(250, 500 m^3/hr), pressure distribution at volute is dense and increase rapidly. Cavitation is seems to be more near blade inward curve side.
3. Pump efficiency is higher at 1000- 1200 m^3/ hr range. Pump performance is satisfactory at our designed condition (1000 $m^{3/}$ hr).
4. Whirl component is more at volute neck, means back flow is observed at neck and impeller-volute interface.
5. Efficiency of the pump likely to be reduced after 1500 m^3/hr. Head also reduced sharply.
6. At lower discharge condition, pump is showing vibration, uneven pressure and velocity distribution.

REFERENCES

[1] Mohamad hazeri Ismail. Design and development of centrifugal pump impeller for performance enhancement. https://www.researchgate.net/publication/285055176

[2] Farah Elida Selamat. Design and Analysis of Centrifugal Pump Impeller for Performance Enhancement. Journal of Mechanical Engineering Vol SI 5(2), 36-53, 2018

[3] David Cowan, Thomas Liebner, Simon Bradshaw. . Influence of impeller suction specific speed on vibration performance. Proceedings of the Twenty-Ninth International Pump Users Symposium October 1-3, 2013, Houston, Texas

[4] Jianhua Liu, Xiangyang Zhao and Miaoxin Xiao. Study on the Design Method of Impeller on Low Specific Speed Centrifugal Pump. The Open Mechanical Engineering Journal*,* 2015, *9,* 594-600

[5] Mukesh Sahdev. Basics Concepts of operation, Maintenance, and Troubleshooting. The Chemical Engineers' Resource Page, www.cheresources.com

[6] Marcus Beck, Paul Uwe Thamsen. How to design a centrifugal pump with constant power consumption for all flow rates Distributed under a Creative Commons Attribution| 4.0 International License https://hal.archives-ouvertes.fr/hal-01549128

[7] R. M. Pande, S. U. Kandharkar, R. B. Patthe, V.M.Nandedkar, V.B. Tungikar. Computational Fluid Dynamics (CFD) of Centrifugal Pump to Study the Cavitation Effect. International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME)

[8] Tilahun Nigussie, Edessa Dribssa Design and CFD Analysis of Centrifugal Pump International Journal of Engineering Research and General Science Volume 3, Issue 3, May-June, 2015

[9] Raghavendra S Muttalli , Shweta Agrawal , Harshla Warudkar , CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX International Journal of Innovative Research in Science, Engineering and Technology

[10] Mr. Nilesh Patil , Prof.G.S.Joshi , Prof.Dr.V.R.Naik Validation in the improved performance of Centrifugal pump using CFD International Research Journal of Engineering and Technology (IRJET)

[11] Mr. Jekim J. Damor, Prof. Dilip S. Patel, Prof. Kamlesh H.Thakkar , Prof. Pragnesh K. Brahmbhatt Experimental and CFD Analysis Of Centrifugal Pump Impeller- A Case Study international journal of engineering and research technology

[12] Mehta Mehul Pravinchandra Improving The Head And Efficiency of A Pump 2016 IJEDR | Volume 4, Issue 2

[13] X Q Zheng et al Study on internal flow field simulation accuracy of centrifugal impellers based on different meshing types IOP Conference Series: Earth and Environmental Science